ANALYZE OF THE WIND LOAD INFLUENCES ON THE SHAPE TALL BUILDING AND SURFACES OPTIMIZATION USING COMPUTER FLUID DYNAMICS METHOD

Lecturer eng. **Ioan-Sorin LEOVEANU,** PhD.[1], Assoc. prof. eng. **Eva KORMANÍKOVÁ,** PhD.[2], Assis. prof. eng. **Kamila KOTRASOVÁ,** PhD.[2]

[1]University „Transilvania" from Braşov,
[2]Technical University of Košice, Civil Engineering Faculty, Institute of Structural Engineering, Department of Structural Mechanics, Košice, Slovakia

REZUMAT. Una dintre problemele majore întâlnite la proiectarea construcțiilor înalte o constitue încărcarea produsă de vânt pe suprafața exterioară a clădirii. De regulă, aceste încărcări sunt stabilite experimental, în tunelele de vint, unde se lucrează pe modele la scară ale clădirii a cărei structură se proiectează, ca și ale celor vecine acesteia, deja existente. Scopul acestei lucrări este acela de determina încărcările produse de vânt asupra clădirilor foarte înalte utilizând metodele numerice de simulare a curgerii aerului în jurul clădirii, ca și determinarea formei geometrice a clădirii care înlătură majoritatea problemelor legate de variațiile mari ale presiunii pe suprafețele exterioare ale clădirii. În cazul analizat, vântul se consideră acționând în rafale, iar în continuarea lucrării, suprafața exterioară a clădirii va fi denumită anvelopă. Modelul de calcul dezvoltat în lucrare se bazează pe metoda volumelor finite, iar modelarea în jurul fazelor solide se face utilizând o procedură dezvoltată pe baza TVD de ordinul 2. Rezultatele acestei metode sunt dezvoltate, în scopul diminuării erorilor, utilizind solverul Riemann și Godunov.

Cuvinte cheie: metoda volumului finit, suprafață de presiune, metoda TVD, construcții înalte, forma optimă a invelișului.

ABSTRACT. One of the principal problems in the process of tall buildings design consists in the evaluation of the wind pressure around the all exterior surfaces of the buildings. That wind loading systems are established by experimental methods, in the wind simulator installation on the reduction models of the current designing structure and his existent neighbors buildings. The aim of that work consist in numerical methods used in the process of pressures maps on tall buildings exterior surfaces established and there optimal exterior shape for avoiding the vortexes and huge vibrations involved by wind dynamics. In the present paper the wind is considered acting in pulsed mode and in the present paper, the exterior surface of the building will be called envelop. The gases transport phenomenon governing equations was solved using volume finite method (VFM) with the total vanished diminishing (TVD) for flow around the envelope modeling. The new CFD VFM TVD algorithm was developed for minimization errors and is based on Riemann and Godunov solver.

Keywords: volume finite method, CFD, surface pressure, TVD method, tall buildings, optimal shape's envelop.

1. INTRODUCTION

The wind loading is considered by Devenport (1998) in three categories:

a) Extraneously-induced loading based on naturally turbulent oncoming wind. The weak of upstream obstructions enhance this categories buffeting.

b) Unstable flow phenomenon such as separations, reattachments and vortex shedding generate a secondary type of forces.

c) The movement-induced excitation of the body generate by the deflection of the structure create fluid flow too. This phenomenon with a strong unsteady states character gives the complexity of the fluid flow around the flexible tall structures. The modern design of flexible tall structures must request to earth quakes events and wind loads, cases that represent a state of the art of the civil engineering.

The Eurocode and most used standards are highlighted by Allsop (2009) as the most flexible and inclusive code for normal buildings. The quasi-static methods offered by these codes are only applicable for buildings with structural properties such that they are not susceptible to dynamic excitation (Metha, 1998).

Thus, the tall buildings, those with high slenderness ratios and/or asymmetric planes, exceed limitations and are advised to be tested in the wind tunnel. This pattern induces inherent structural flexibility and heightens concerns regarding the aero-elastic fluid-structure interaction between the wind and the tall building. The codes of practice have been formulated with a view to providing an acceptable balance between the overly complex reality and oversimplified approach. Scaled-model wind tunnel testing is an established tool among industry design practices. Boundary layer wind tunnels are capable of quantifying time-dependent surface pressures, including the complex types of loadings (torsion and across wind). The model can be used to determine the best orientation of the proposed building, the case of Burj Dubai analysis (Irwin and Baker, 2006). The simulation, nevertheless, it has its own limitations that include the difficulty to maintain proportionality between the scaled turbulence characteristics and the scaled building model, especially if the topography is significant (Taranath 1998). Furthermore, it is important to ensure Reynolds number effects on the pressures are kept to a minimum. It is noted by Sun *et al*. (2009) that a computational approach has the capability of being more flexible than traditional wind tunnel experiments. For example, a fully coupled solution between computational fluid dynamics (CFD) and finite element modelling (FEM) can be developed to model the fluid-structure interaction (FSI). A wind tunnel test relies on the simplified assumption that the scaled aeroelastic model can satisfactorily replicate the dynamic properties of the full-scale design. It neglects the influence of higher modes.

The application of CFD for practical wind engineering problems has received a lot of research attention over the last three decades and has made major progress due to the advancement of computer technology. Thus far, the leading applications for the built environment have concentrated on mean wind speeds for areas including: natural ventilation; pollution dispersion; and human comfort at street and balcony level (Stathopoulos, 1997). It has proven very difficult for CFD to acceptably model the complex flow interference phenomena induced from buildings.

Typical features of this unsteady flow regime include turbulent length scales and separation regions larger than the body size of the structure. This is the reason less work has been performed on predicting time-dependent surface pressures on these man-made bluff bodies. CFD has not developed enough to suggest it could replace wind tunnel testing in this respect. It does, however, offer encouraging potential to act as a complimentary tool. In this paper, the various turbulence models will be discussed with respect to their ability to predict surface pressures and resulting wind loads for a tall building. This includes a detailed review of previous validation studies performed within the literature. It proceeds to highlight the user-defined criteria that must also be satisfied. Finally, the scope for future research on simplifying CFD analyses for tall buildings is discussed, with a view to producing a more efficient and practical solution.

The modern simulation is based on turbulence models, based on different models of turbulence used for solving the conservation of mass, momentum and energy equations. Sun et al., 2009 and Castro, 2003 use different CFD turbulence model for analyze the wind loads on the tall buildings. The selection of the turbulence model is made based on accuracy, computational cost, accessibility and available time for simulations. The most complete form of CFD is the Direct Numerical Simulation (DNS) method that uses the direct solution of Navier-Stokes equations for each control volume. The disadvantage of this method consists in the mesh size dimension conditions. The cell mesh must be smallest that the vortex eddy within the flow for capturing the turbulent effects. So, the cost of DNS become extremely high and Knapp (2007) conclude that the DNS method should be limited by a small scale simulation and low Reynolds numbers. The other methods are implemented in ANSYS Inc, (2005) and is based on Reynolds-averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) and are the two most used method for wind load simulations.

The RANS methods are based on the two popular most used models, *k*-e and *k*-w, where k represent the kinetic energy and e the turbulent dissipation rate or w the specific dissipation rate. The RANS method is based on additional empirical equations for establish the turbulent viscosity and are relatively simple to use and robust and can describe the full spectra of turbulence scale. Other methods are based on transient solutions based on spatial filtering approach adapted with subgrid-scale for smallest cell dimension eddy. The mesh in that method must be smallest that in the RANS method but the filtering approach can give more information about turbulence areas of the model. In the last years, was developed a new method based on the RANS and LES models, named Detached Eddy Simulation (DES), method where the simplest RANS algorithms are used for majority flow domains simulate and LES is used only in the area of separated flow. The use of DES method in the wind load estimation of tall structures consists in the high turbulent area developed around the buildings. That shortest actual situation represents the consideration for using the CFD methods for shape envelope consideration.

2. NUMERICAL ANALYSIS

In this paper we establish the pressure distribution on the façades of the tall buildings using the

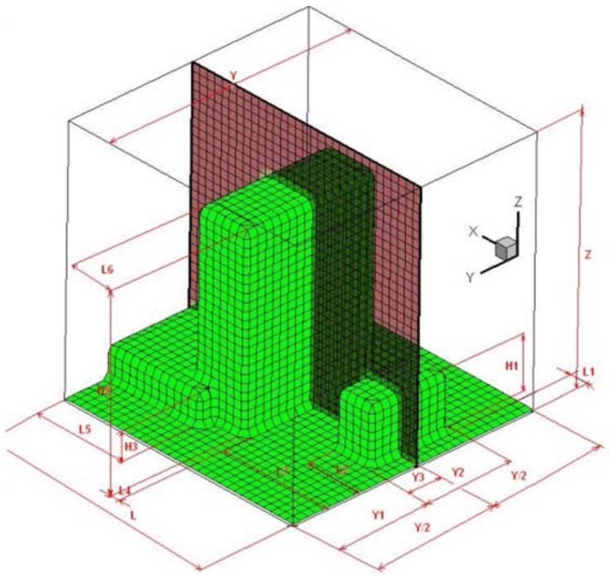
TVD algorithms for gas dynamics based on Euler PDE system of equation for a situation of wind loading more closely of the reality. The domain of computing is established in the Figure 1 and the wind input speed diagram in the Figure 2. The system of partial differential equations is give on the forms of mass conservation (*1*), momentum conservation (*2*) and energy conservation (*3*). The expression of that system of partial equations is done in Table 1.

The system of equations is writes as:

$$q = f_{(q)} + g_{(q)} + h_{(q)} \tag{4}$$

Table 1. **The transport equations used for the wind pressure loadings dynamics**

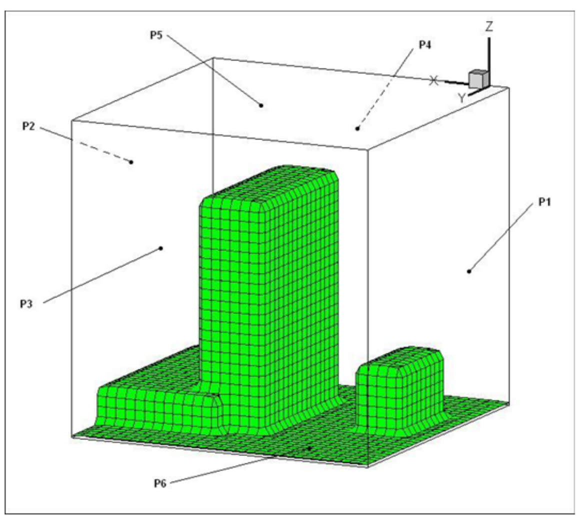
$\frac{\partial \rho}{\partial t} + \nabla\left(\rho \cdot \vec{V}\right) = 0$	$\frac{\partial}{\partial t}\left(\rho \cdot \vec{V}\right) + \nabla\left(\rho \cdot \vec{V} \times \vec{V}\right) = -\nabla p + \rho \cdot \vec{g}$	$\frac{\partial}{\partial t}(\rho \cdot E) + \nabla(\rho \cdot E + p) \cdot \vec{V} = \nabla(\lambda \cdot \nabla T)$
(1)	(2)	(3)

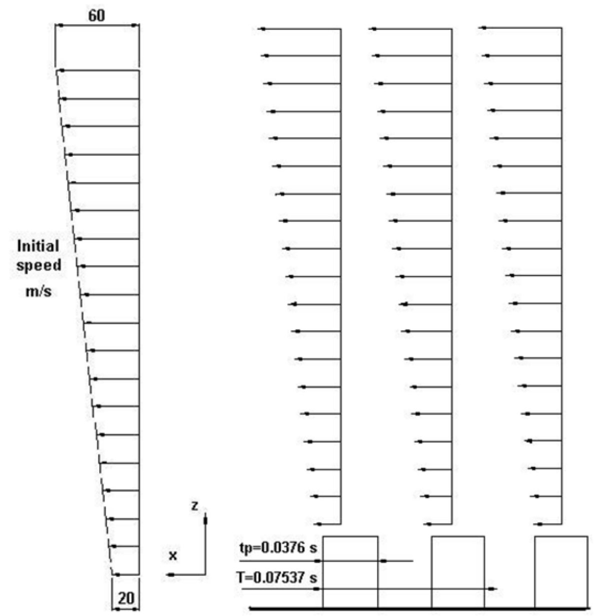


Index *Ox*	Value [m]	Index *Oy*	Value [m]	Index *Oz*	Value [m]
L	96	*Y*	96	Z	96
L1	6	*Y*/2	48	H1	21
L2	21	*Y*1	36	H2	69
L3	46	*Y*2	33	H3	12
L4	3	*Y*3	24		
L5	36	*Y*4	24		
L6	18				

a) b)

Fig. 1. Geometrical domain dimensions.





a) b)

Fig. 2. The plane notation of the domain analyzed, a) and the inlet speed distribution on *Ox* direction.

where:

$$q=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ E\end{bmatrix};\ f_{(q)}=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ \rho uw\\ u(E+p)\end{bmatrix};$$

$$g_{(q)}=\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ \rho vw\\ v(E+p)\end{bmatrix};\ h_{(q)}=\begin{bmatrix}\rho w\\ \rho uw\\ \rho vw\\ \rho w^2+p\\ w(E+p)\end{bmatrix} \quad (5)$$

The total energy E represents the sum of internal and kinetic energy and is expression is:

$$E=\frac{p}{\gamma-1}+\frac{1}{2}\rho\left(u^2+v^2+w^2\right) \quad (6)$$

and the equation of state for a g-law polytrophic gas considered in the present work has the shape

$$\gamma=\frac{c_p}{c_v} \quad (7)$$

where c_p and c_v are the specific heat at constant pressure, respectively constant volume.

The geometry of tall building and his vicinities buildings used in present work is done in the Figure 1. Geometrical dimensions of solid area are done in Figure 1*b*.

3. BOUNDARY AND INITIAL CONDITIONS AND INPUT PARTICULARITIES

The Boundary conditions used in the solving problems consist in free output from the planes P2, P3, P4 and P5, as in figure 2a and in input from plane P1. The inlet gas in the computation domain is made from plane P1 with a speed distribution as in figure 2b. The speed distribution is made in accord with the maximum speed at 60 year measurement on wind in the most affected area on India, in agreement with the design norms for wind load. The speed distribution on Ox is considered linear on all the inlet area for verification the turbulence that can appear in the fluid trap between the two buildings with different height. The inputs from plane P1 have a pulse shape with equal time between the pulse duration and pause between two pulses.

4. ANALYSIS RESULTS AND OPTIMIZATION REASONS

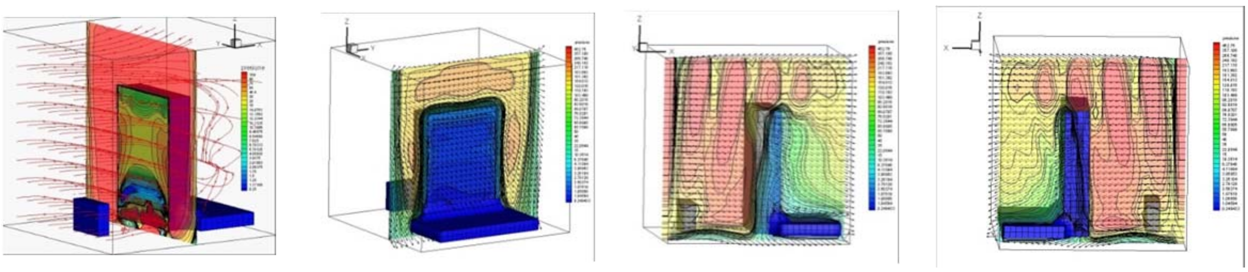
The results of the CFD analysis from actual geometry is presented in the Figure 3 for pressure in Figure 4 for speed on Ox, (u), for some time intervals:

The results obtained from initial geometrical configurations put in relief a strong vortex area between the tall building, (69 m, 23 levels) and the normal building, (21 m height 7 levels) as is presented in Figure 5.

5. SHAPE OPTIMIZATION PROCESS

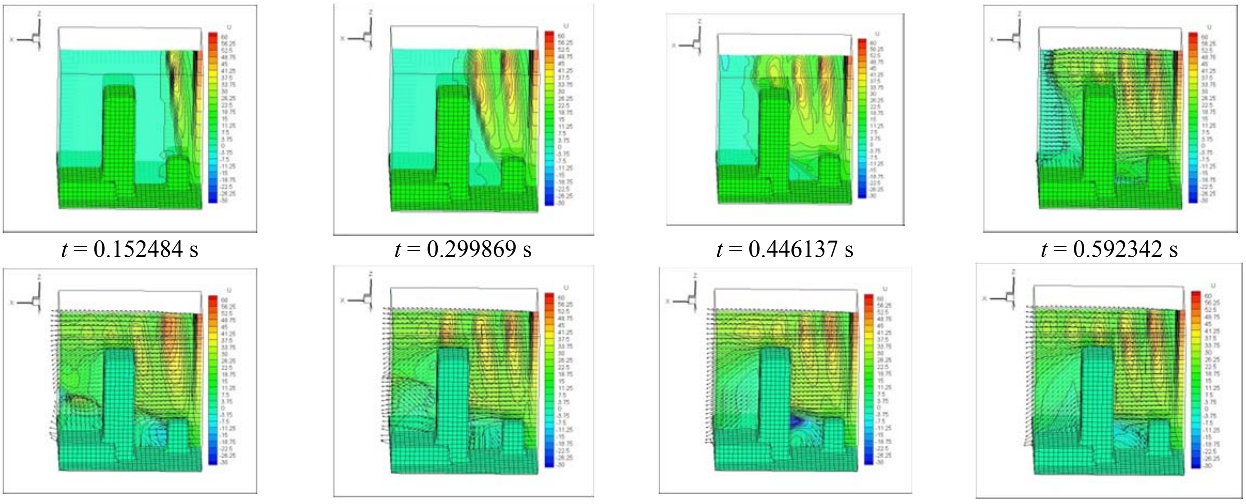
The optimization process consist in an iterative running application that modify the shape of the envelope based on the maximum variation pressure on the faces of boundary volumes finite that meshes the tall building.

The maximum pressure variation is determinate using the monitoring of the fluid pressure on each volume finite area for every time step of simulation. We will determine the maximum pressure P_{max} and minimum pressure P_{min} that appear on all time of simulation. The maximum pressure variation is compute fore every area of finite volume in contact with the tall building envelope. The Finite Volumes that have a value of DP_{max} great that an imposed accepted AD_{Pmax} will be eliminate from the analyze and the boundary conditions will be determinate according with that new situation.



a) b) c) d)

Figure 3. The pressure maps on the lateral surfaces of tall building:
a – the streams line of the gas flow and the pressure in the façade section at time 5.39784 s of simulation; *b* – the pressures on the back surface of the building; *c*, *d* – the pressure on the right and left sites of the tall building.



t = 0.152484 s | t = 0.299869 s | t = 0.446137 s | t = 0.592342 s

t = 0.731194 s | t = 0.871237 s | t = 1.534835 s | t = 5.39784 s

Fig. 4. Gas speed U [m/s] in the xOz symmetry plane of the domain of analyze for different time moments.

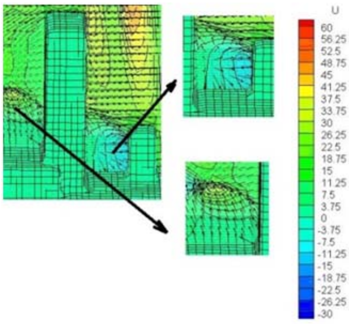
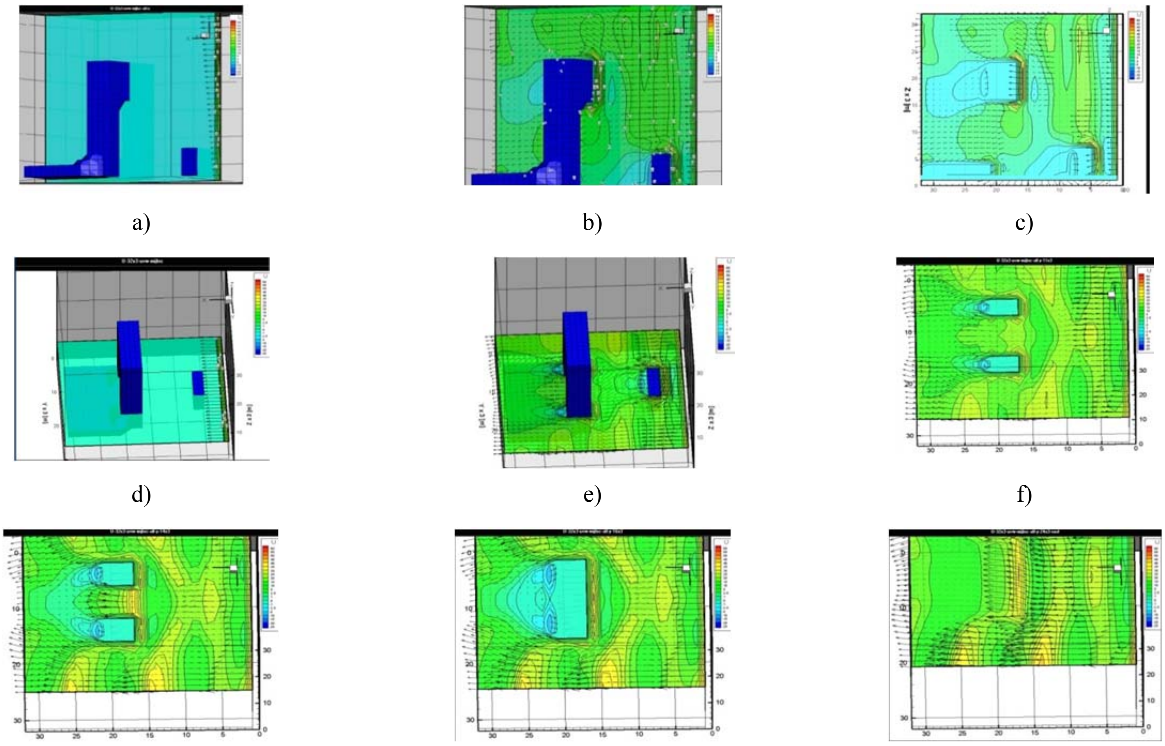


Fig. 5. The vortex area of interest in the process of envelope modification.



a) b) c)

d) e) f)

g) h) j)

Fig. 6. The results of optimization process:
a – the speed input and final shape generated; *b*, *c* – the speed on *Ox* axis in the middle section of the domain at final time simulation, t = 5.39784 s; *d* – the upper view of the computing domain resulted from the optimization process; *e* – the speed on *Ox* axis in the section at 18 m elevation; *f* – the speed on *Ox* axis in the section at 33 m elevation; *g* – the speed on *Ox* axis in the section at 42 m elevation; *h* – the speed on *Ox* axis in the section at 48 m elevation; *j* – the speed on *Ox* axis in the section at 72 m elevation.

The simulation process is reloaded for the new shape envelope and other Finite Volumes will be input or extract from the domain geometry. When the pressure variation will be little that the accepted pressure variation or the number of iteration exceeds the imposed maximum iteration the process is stopped.

The computing algorithms method for wind loading was developed in Microsoft Visual C/C++ and the solvers results was plot using TecPlot 360 software utility.

6. CONCLUSIONS

The gas dynamics modelling based on Euler PDE system of equation can solve the problems of wind loads on tall structures without using the Navier-Stokes PDE system of equation with diverse turbulence flow models for accurate flow dynamics.

A combination between the two modelles can be used because the gas speeds are low in the case of wind loads and the gas is practical incompressible. The turbulence area of flow, that in the civil engineering have a huge area of the domain (60-80%) in the cases of wind loads on tall buildings can be simulate using the Euler system of equations and the complicated turbulences models can be avoided.

In the present paper was proposed a new optimization method from establish the optimum envelope of a tall building. The results obtained after the optimization process presented in the Figure 6 for the final time analyze (t = 5.39784 s) assure the huge reduction of maximum pressure variation and the vibrations of the structure building in the case of wind action.

The building comfort indices assured by the process of optimization suggest a chipper dynamics adsorption of vibration.

The computing algorithms method for wind loading was developed in Microsoft Visual C/C++ and the solvers results was plot using TecPlot 360 software utility.

REFERENCES

[1] Allsop, A. *BS EN 1991-1-4 Tall Buildings*. In: ICE (Institution of Civil Engineers), New Eurocode on Wind Loading (London, UK, 11 May 2009).

[2] ANSYS Inc. *FLUENT 12.0 Theory Guide*. (2009)

[3] Building Research Establishment. *Wind around tall buildings*, BRE Digest 390, (BRE: Watford, UK, 1994).

[4] British Standards Institution. *BS 6399-2:1997 Loadings for buildings – Part 2: Code of practice for wind loads.* (BSI: London, UK, 1997).

[5] British Standards Institution. *BS EN 1991-1-4:2005 Eurocode 1: Actions on structures – Part 1-4:General actions – Wind actions.* (BSI: London, UK, 2005).

[6] Castro, I.P. *CFD for External Aerodynamics in the Built Environment.* The QNET-CFD Network Newsletter. Vol. 2: No. 2 – July 2003, pp 4-7 (2003).

[7] Davenport, A.G. *What makes a structure wind sensitive?* in Proc. The Jubileum Conference on Wind Effects on Buildings and Structures. 8. Porto Alegre, Brazil 25-29 May 1998, eds. A. A. Balkema, (Rotterdam, Netherlands, 1998) pp 1-14.

[9] Irwin, P.A. and Baker, W.R. *The Burj Dubai Tower Wind Engineering.* Structure Magazine, June 2006 pp 28-31 (2006).

[10] Knapp, G.A. *Improved methods for structural wind engineering*. Ph. D. University of Nottingham(2007).

[11] Nozu, T., Tamura, T., Okuda, Y., and Sanada, S. *LES of the flow and building wall pressures in the center of Tokyo.* Journal of Wind Engineering and Industrial Aerodynamics, Elsevier, **96** pp 1762-1773 (2008).

[12] Leoveanu, I.,S., s.a., *Metode numerice avansate. Aplicatii in modelarea metalelor.* Ed. Universității Transilvania din Brașov, 2009.

[13] Vizman, D.; Cristea, A.; Sofonea, V. *Metode numerice avansate.* Eurobit. Timişoara, 2008.

[14] Leoveanu, I. S., Kotrasova, Kamila. *Numerical modelling of ignition in TNT deflagration.*

[15] Leoveanu, I. S., et al. *Computer fluid dynamics application for establish the wind loading on the surfaces of tall buildings.* In: Proceedings of the 13th WSEAS international conference on Automatic control, modelling & simulation. World Scientific and Engineering Academy and Society (WSEAS), 2011. p. 433-438.

[16] Janas, Petr; Krejsa, Martin; Krejsa, Vlastimil. *Software package ProbCalc from the point of view of a user.* Transactions of the VŠB–Technical University of Ostrava, Civil Engineering Series, 2010, 10.1: 1-11.

[17] Králik, Juraj. *Probability assessment of analysis of high-rise buildings seismic resistance.* Advanced Materials Research, 2013, 712: 929-936.

[18] Krejsa, Martin; Janas, P. E. T. R.; Krejsa, Vlastimil. *Direct Optimized Probabilistic Calculation.* In: Recent Advances in Systems Science & Mathematical Modelling: Proceedings of the 3rd International Conference on Mathematical Models for Engineering Science (MMES'12). 2012. p. 216-221.

[19] Krejsa, Martin; Janas, Petr; Krejsa, Vlastimil. *Application of the Direct Optimised Probabilistic Calculation Method.* In: Recent Advances in Applied and Theoretical Mathematics: Proceedings of the 18th WSEAS International Conference on Applied Mathematics (AMATH'13). p. 200-205.

[20] Hong, C.,P., *Computer Modelling Of Heat And Fluid Flow In Material Processing.* Editor: *IOP PUB*, ISBN: 0 7503 0445 6.

[21] Leoveanu, I., S. *Advanced Numerical Method Used In Computer Fluid Dynamics Modellings Of Loading On Buildings.* In: https://www.researchgate.net/publication/; doi: 10.13140/2.1.5088.4165.

Despre autori

Lecturer dr. eng. **Ioan Sorin LEOVEANU**
University „Transilvania” from Braşov

Mechanical Engineer of the University Transilvania from Brasov, the Managerial Industrial Program with Welding Especiality and PhD in residual stresses and strains modelling and technology optimisation. He worked at the Industrial Tractors Design and Research Institute at ICPATT Brasov to havy and mediun Bulldozers prototips design and other Earth Moving Machineries prototipes and series products. From 1988 he work at Transilvania University at Materials Science and Engineering Faculty and from 2010 he work in the area of Civile Engineering at Transilvania University. He publish monographis in the area of Optimization Technology and Transport Phenomenon involved in the Welding and Engineering area and articles in diverses journals and national and international conferences. Here research topics. Modelling the physical processes involved in welding phenomenons using Finite Volume and Finite Elements Method for Modelling the Exceptional Loads induced in Builings by Earth Quakes, Wind and Explosions.

Assoc. prof. eng. **Eva KORMANÍKOVÁ**, PhD
Technical University of Košice, Civil Engineering Faculty, Institute of Structural Engineering, Department of Structural Mechanics, Košice, Slovakia

Graduated at the Technical University of Košice, Civil Engineering Faculty, study program - Building Construction. After finishing of the university she started to work at the Technical University of Košice, Civil Engineering Faculty, Department of Structural Mechanics as assistant. PhD. graduated at the Technical University of Košice, Faculty of Mechanical Engineering, study program – Applied Mechanics. Since 2009 she has worked at Civil Engineering Faculty TUKE, study program – Theory and Design of Engineering Structures, as associate professor. Her research topic is design and optimization of structural elements and structures made of composite materials.

Assis. prof. eng. **Kamila KOTRASOVÁ**, PhD
Technical University of Košice, Civil Engineering Faculty, Institute of Structural Engineering, Department of Structural Mechanics, Košice, Slovakia

Graduated at the Technical University of Košice, Civil Engineering Faculty, study program - Building Construction. After finishing of the university she started to work at RCB in Spišská Nová Ves as designer and then at the Technical University of Košice, Civil Engineering Faculty, Department of Structural Mechanics as assistant.
PhD. graduated at the Technical University of Košice, Faculty of Mechanical Engineering, study program – Applied Mechanics. The research topics: seismic design of liquid storage ground-supported tanks, interaction problems of fluid and solid.